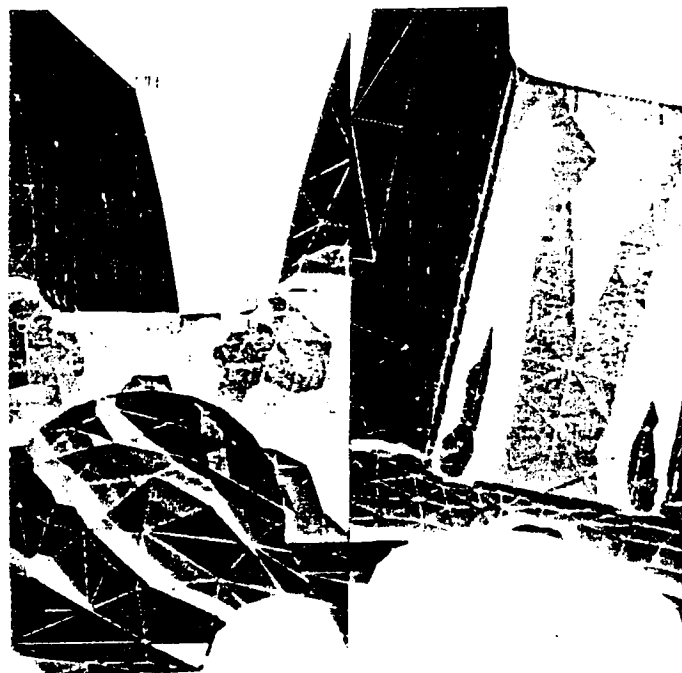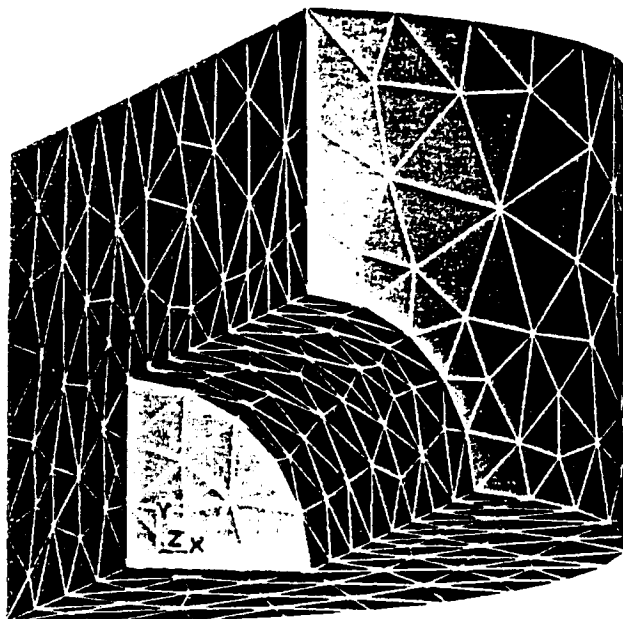# Finite Element Meshing of ANSYS® Solid Models

F. S. Kelley, Supervisor, General Development
Swanson Analysis Systems, Inc.
P. O. Box 65, Johnson Road
Houston, Pennsylvania 15342
Telephone: (412) 746-3304

Presentation at the Workshop on the Integration of
Finite Element Modeling with Geometric Modeling
Rennselaer Polytechnic Institute
May 1987

(All images are from ANSYS Revision 4.3)

1. INTRODUCTION - ANSYS AND SASI HISTORY

Swanson Analysis Systems, Inc. (SASI) was founded in 1970 by Dr. John A. Swanson to develop, support, and market ANSYS, a large scale, general purpose finite element computer program. ANSYS and the recently introduced ANSYS-PC products remain the only reasons for SASI's existence. There is no engineering consulting practice to distract attention away from the software business. SASI currently employs over 100 people at its office near Pittsburgh, Pennsylvania, and there are thirty regional support distributors marketing and supporting ANSYS worldwide.

ANSYS was developed solely for the commercial market, with no government or university funding. It has more than 1000 installations to date, including universities, but not PC's.

2. PURPOSE OF ANSYS SOLID MODELING

ANSYS was perhaps the first commercially available program to offer truly interactive finite element model generation. (In the late 1970's, there was confusion about what constitutes "interactive" processing. Some programs would simply prompt users for a fixed sequence of commands.) ANSYS Revision 3, released in August 1978, contained PREP7. This processor allowed a user to create, display, and modify a finite element mesh in whatever order desired.

ANSYS Revision 3 also contained a powerful 3-dimensional automatic mesh generator called PREP5. Based on keypoints, lines, areas, and volumes, this processor created brick models with relative ease. It was also capable of automatic application of boundary conditions. PREP5 was never as popular for model creation as PREP7. However, some users were upset when we removed PREP5 at Revision 4.0, released in 1982.

ANSYS Revision 4.0 (1982) introduced the PREP7 Mesh module, with powerful automatic quadrilateral and brick meshing capabilities. The 4.0 MESH module was widely used for model generation, but it could not handle irregular regions.

The ANSYS PREP7 MESH module was rewritten as a solid modeler for Revision 4.2 (1985), and enhanced in Revision 4.3 (to be released in June 1987). This was done solely to aid ANSYS users in the creation of finite element analysis models. SASI did not have to patch finite element meshing into the ANSYS solid modeler as an afterthought. It was designed in from the beginning.

From SASI's point of view, any other benefits which may be derived from the creation of a solid model in ANSYS (such as pretty pictures) are bonuses rather than primary objectives.

3. ANSYS REVISION 4.3 SOLID MODELS

ANSYS solid models are internally stored in several forms. The first of these has been well documented in textbooks and papers. Lines, surfaces, and volumetric regions are defined by Hermite cubic splines as shown below. The parameters r, s, and t vary from 0.0 to 1.0. See figures 1 - 3.

$$X = C1 + C2 \cdot r + C3 \cdot r^2 + C4 \; r^3 \qquad \text{(line)}$$
$$Y = C5 + C6 \cdot r + C7 \cdot r^2 + C8 \; r^3$$
$$Z = C9 + C10 \cdot r + C11 \cdot r^2 + C12 \; r^3$$

$$X = C1 + C2 \cdot r + C3 \cdot r^2 + C4 \; r^3 \qquad \text{(surface)}$$
$$+ (C5 + C6 \cdot r + C7 \cdot r^2 + C8 \; r^3)\,(s)$$
$$+ (C9 + C10 \cdot r + C11 \cdot r^2 + C12 \; r^3)\,(s^2)$$
$$+ (C13 + C14 \cdot r + C15 \cdot r^2 + C16 \; r^3)\,(s^3)$$
$$Y = C17 + C18 \cdot r + C19 \cdot r^2 + C20 \; r^3$$
$$+ (C21 + C22 \cdot r + C23 \cdot r^2 + C24 \; r^3)\,(s)$$
$$+ (C25 + C26 \cdot r + C27 \cdot r^2 + C28 \; r^3)\,(s^2)$$
$$+ (C29 + C30 \cdot r + C31 \cdot r^2 + C32 \; r^3)\,(s^3)$$
$$Z = C33 + C34 \cdot r + C35 \cdot r^2 + C36 \; r^3$$
$$+ (C37 + C38 \cdot r + C39 \cdot r^2 + C40 \; r^3)\,(s)$$
$$+ (C41 + C42 \cdot r + C43 \cdot r^2 + C44 \; r^3)\,(s^2)$$
$$+ (C45 + C46 \cdot r + C47 \cdot r^2 + C48 \; r^3)\,(s^3)$$

$$X = C1 + C2 \cdot r + C3 \cdot r^2 + C4 \; r^3 \qquad \text{(volumetric region)}$$
$$+ (C5 + C6 \cdot r + C7 \cdot r^2 + C8 \; r^3)\,(s)$$
$$+ (C9 + C10 \cdot r + C11 \cdot r^2 + C12 \; r^3)\,(s^2)$$

$$+ (C13 + C14 \cdot r + C15 \cdot r^2 + C16 \ r^3) \ (s^3)$$
$$+ [C17 + C18 \cdot r + C19 \cdot r^2 + C20 \ r^3$$
$$+ (C21 + C22 \cdot r + C23 \cdot r^2 + C24 \ r^3) \ (s)$$
$$+ (C25 + C26 \cdot r + C27 \cdot r^2 + C28 \ r^3) \ (s^2)$$
$$+ (C29 + C30 \cdot r + C31 \cdot r^2 + C32 \ r^3) \ (s^3) \ ] \ [t]$$
$$+ [C33 + C34 \cdot r + C35 \cdot r^2 + C36 \ r^3$$
$$+ (C37 + C38 \cdot r + C39 \cdot r^2 + C40 \ r^3) \ (s)$$
$$+ (C41 + C42 \cdot r + C43 \cdot r^2 + C44 \ r^3) \ (s^2)$$
$$+ (C45 + C46 \cdot r + C47 \cdot r^2 + C48 \ r^3) \ (s^3) \ ] \ [t^2]$$
$$+ [C49 + C50 \cdot r + C51 \cdot r^2 + C52 \ r^3$$
$$+ (C53 + C54 \cdot r + C55 \cdot r^2 + C56 \ r^3) \ (s)$$
$$+ (C57 + C58 \cdot r + C59 \cdot r^2 + C60 \ r^3) \ (s^2)$$
$$+ (C61 + C62 \cdot r + C63 \cdot r^2 + C64 \ r^3) \ (s^3) \ ] \ [t^3]$$

(The equations for Y and Z are similar, using C65 through C192.)

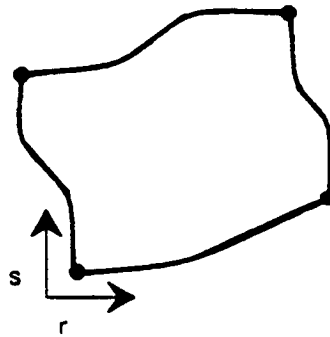Figure 1   Hermite Spline Defining a Line

Figure 2   Bicubic Hermite Spline Defining a Surface Region
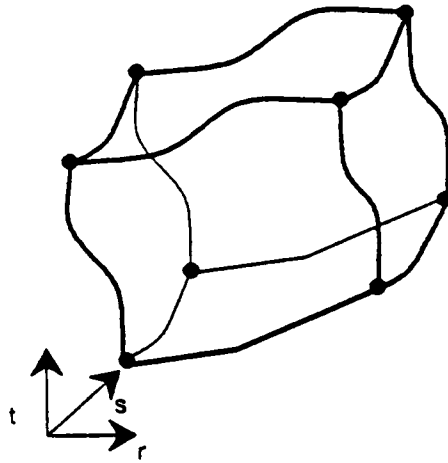
Figure 3   Tricubic Hermite Spline Defining a Volumetric Region

ANSYS also allows the definition of degenerate Hermite regions (figures 4 and 5). This is very important. There is no assurance that an arbitrary surface can be mapped by quadrilateral regions, and even less assurance that an arbitrary 3-dimensional object can be mapped by hexahedral regions. The degenerate forms give ANSYS a far more general modeling capability than would be provided by the standard regions.
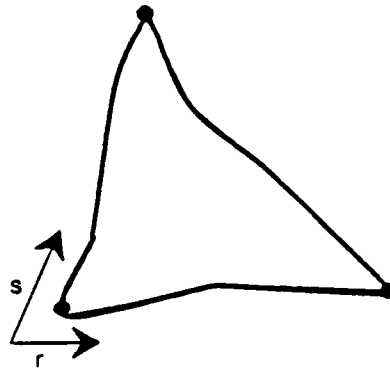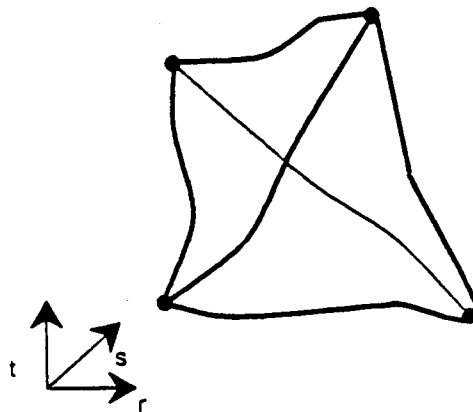

Figure 4   Degenerate Surface Region

Figure 5   Degenerate Volumetric Region

New to ANSYS Revision 4.3 is the ability to define surface regions by a list of up to 200 cubic line segments, and volumetric regions by a list of up to 200 bicubic surface regions (figures 6 and 7). These alternate region types allow great flexibility in the modeling of complex structures. They also make it difficult to classify the ANSYS solid modeler into one category, such as "B-rep" or "CSG". Perhaps "hybrid CSG" is the best term to apply.
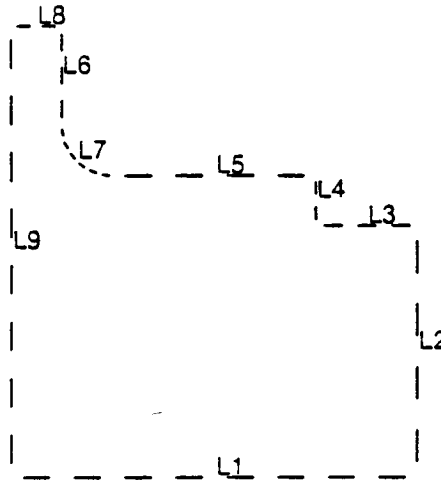

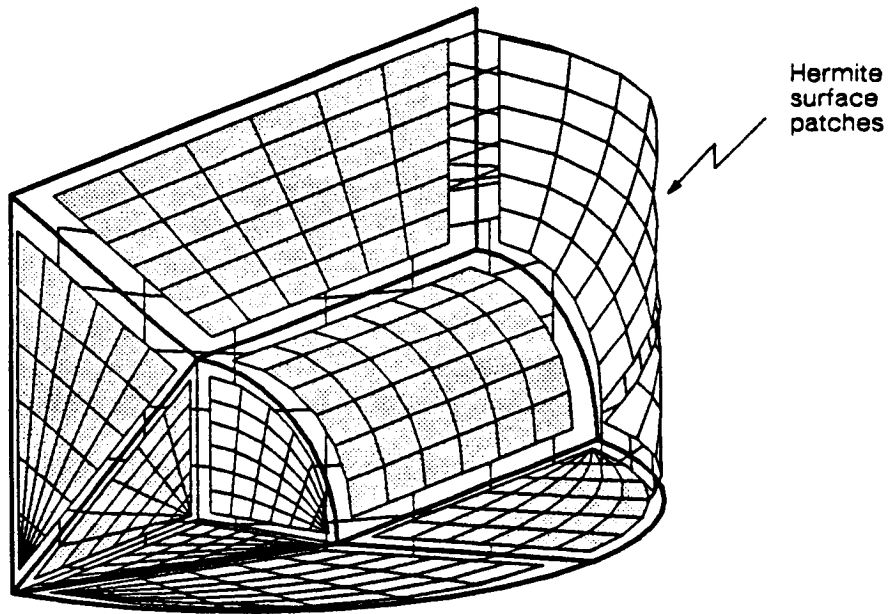Figure 6   Surface Region Defined by a List of Lines

Figure 7   Volumetric Region Defined by a List of Surface Patches

Definition of solid models in ANSYS begins with the input of several "keypoints". "Line segments", "areas", and "volumes" may be defined by connecting keypoints. Lower order entities are generated automatically as needed. Lines and areas follow the curvature of the "currently active coordinate system" (figure 8). Translation, rotation, and symmetry reflection operations are available. Line segments may be rotated about an axis or dragged along a path to produce areas (figure 9). Areas may be rotated about an axis or dragged along a path to produce volumes

5

(figure 10). ANSYS is command driven, with complete documentation available on-line via a menu system. Cross hair and digitizing tablet input is also possible.
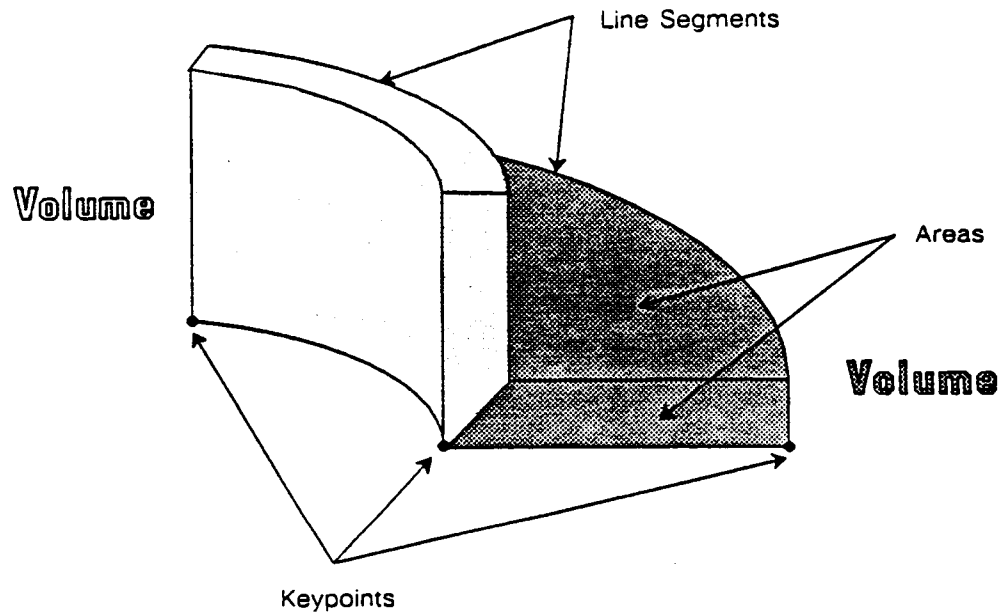

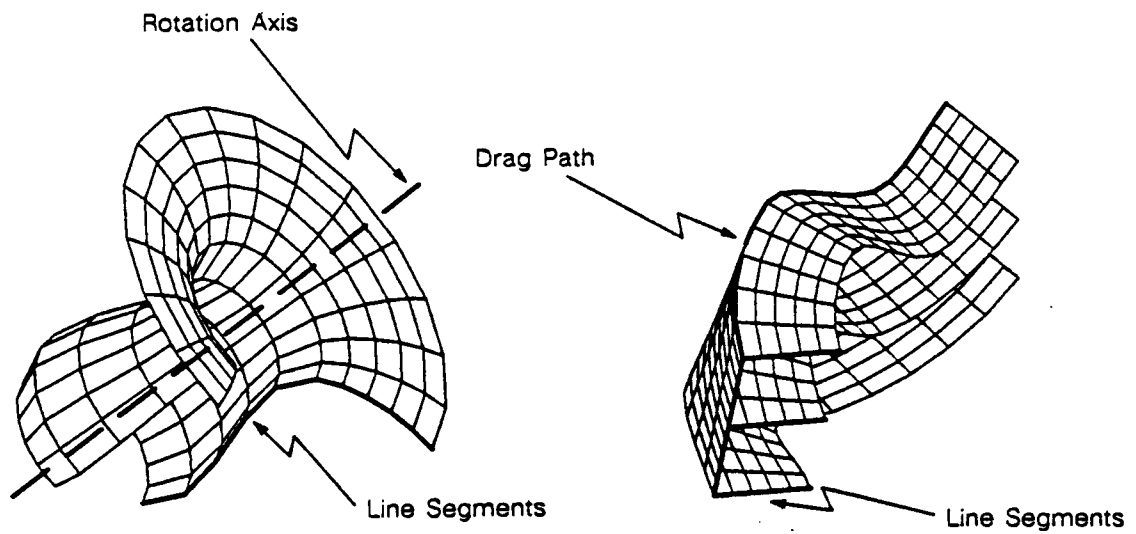Figure 8   ANSYS Keypoints, Line Segments, Areas, and Volumes

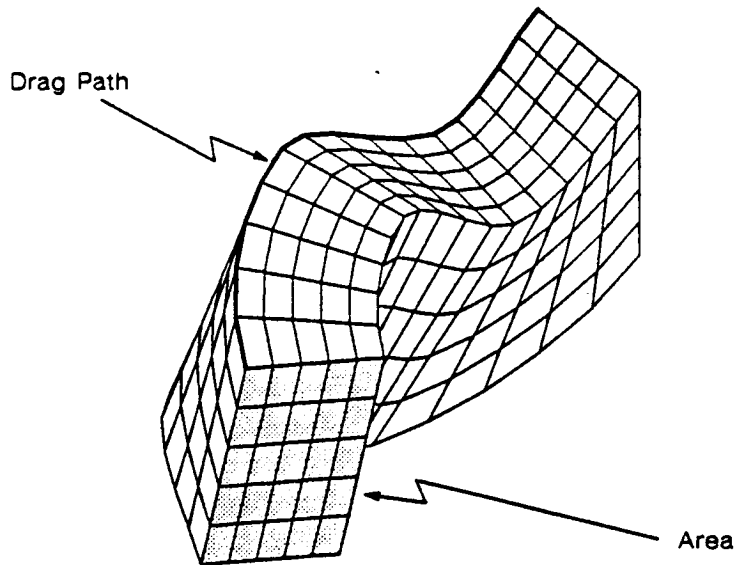Figure 9   Rotation and Dragging of Line Segments to Create Areas

Figure 10   Dragging of an Area to Create Volumes

4. SURFACE ACCURACY OF ANSYS SOLID MODELS

Accuracy of curved surfaces in cubic spline based solid modelers can be of concern. Circular arcs and intersection lines cannot be represented exactly by Hermite splines. A circular arc of 90 degrees has a radius error of 0.03 % (figure 11). For an arbitrary region extending 90 degrees on the surface of a cylinder, the radius error can be as much as 0.2 % (figure 12). Lines resulting from the intersection of arbitrary 90 degree regions can have a radius error of 0.4 % (figure 13). Figure 14 shows the effect of a +/- 0.5 % local perturbation of radius on the results of the analysis of a flat plate with a hole. The maximum corner stress decreased by 0.8 % as a result of the perturbation. For solid elements, the stress error appears to be of the same order of magnitude as the geometric error. Figure 15 shows the effect of a +/- 0.5 % local perturbation of radius on the results of the analysis of a pressurized spherical shell. The stress error introduced was approximately 8 %. For shell elements, the stress error appears to be an order of magnitude higher than the geometric error. The radius error in the ANSYS solid modeler is drastically reduced if the line segments and areas are limited to spans of 45 degrees or less. Typical radius errors are then 0.0005 % for line segments, 0.005 % for areas, and 0.03 % for intersection lines.
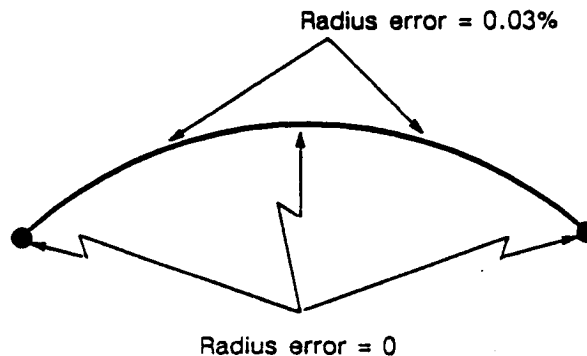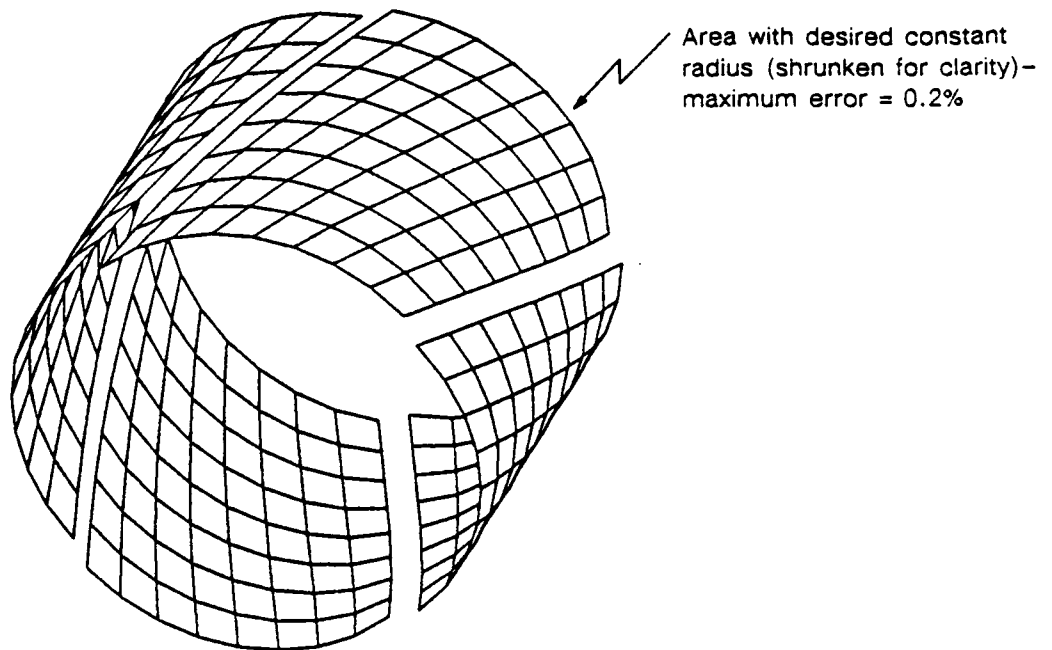

Radius error = 0.03%

Radius error = 0

Figure 11   Radius Error for a 90 Degree Line Segment

Area with desired constant
radius (shrunken for clarity)-
maximum error = 0.2%

Figure 12   Radius Error for Arbitrary 90 Degree Area

Intersection of cylinders –
Maximum radius error = 0.4%
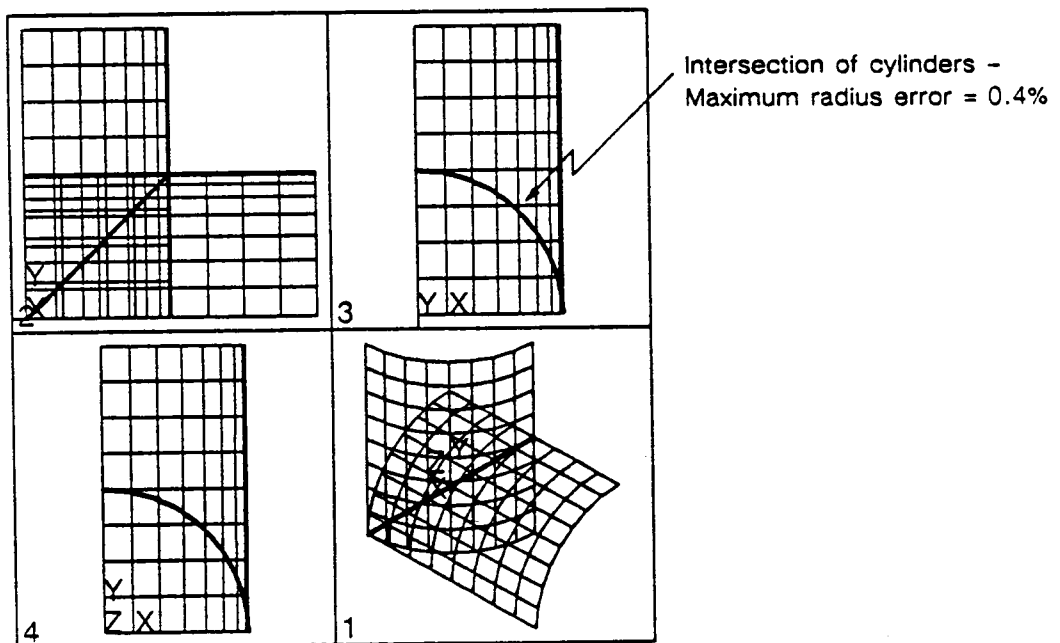
Figure 13   Radius Error for Intersection of 2 Arbitrary 90 Degree Areas

Stress Error = 0.8%

Radius = Constant

Radius varies +/- 0.5%
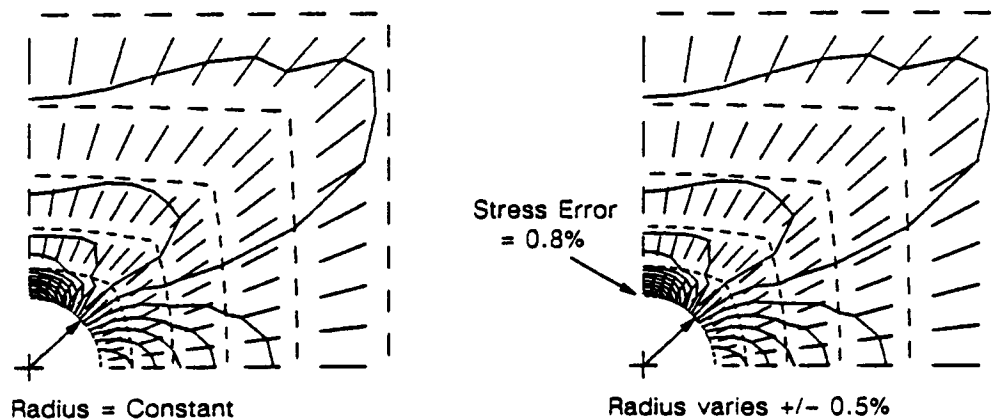
Figure 14   Effect of Radius Error on Plane Stress Solution

Membrane Stress varies +/- 0.5%

Membrane Stress varies +/- 8%

Radius = Constant
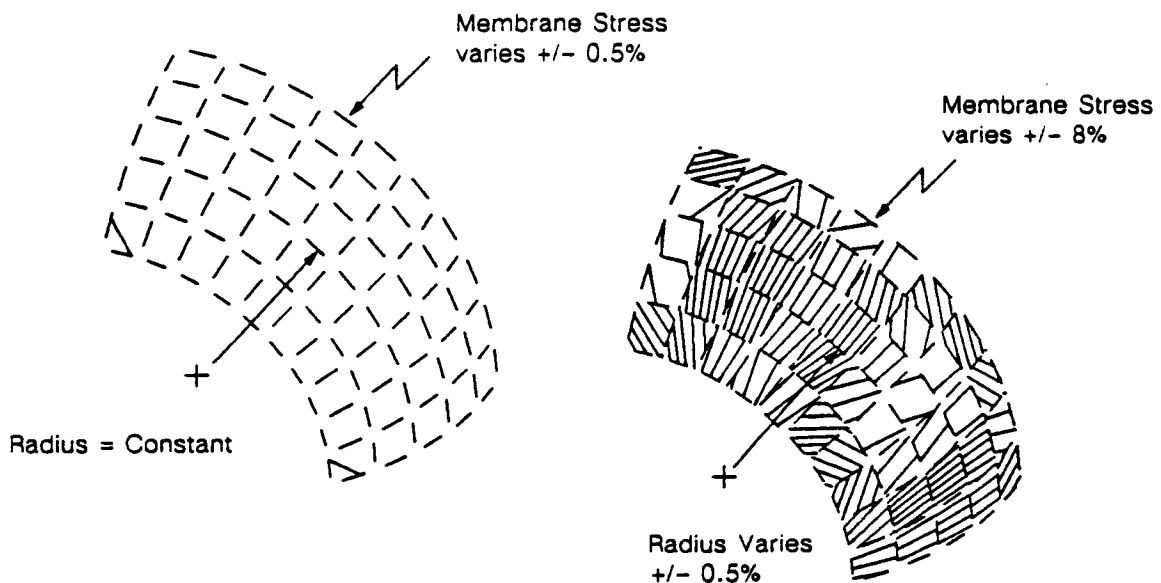
Radius Varies +/- 0.5%

Figure 15   Effect of Radius Error on Shell Stress Solution

5. FINITE ELEMENT MESHING OF AN ANSYS SOLID MODEL

The first step in meshing of an ANSYS solid model is to establish the mesh density. This is accomplished by assigning a number of element divisions and a spacing ratio to every line segment attached to the areas or volumes (figure 16). Commands are available for making the assignments line segment by line segment or to a group of line segments at once. Divisions can be computed based on line segment length and a desired element size and assigned automatically. Spacing ratios can also be computed automatically for smooth mesh transitioning (figure 17).

Divisions Established
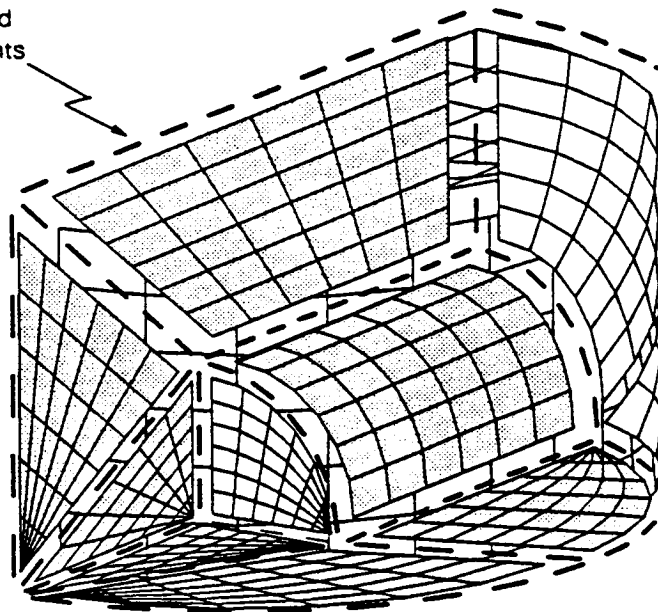for All Line Segments

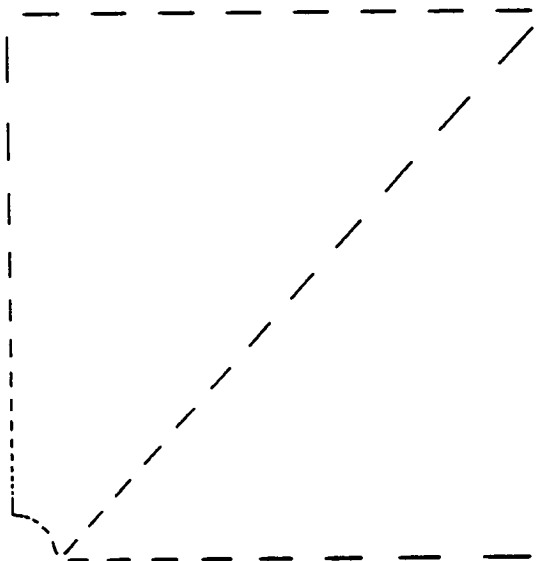Figure 16   Establishing Finite Element Mesh Density

Figure 17   Automatically Adjusted Line Divisions and Spacing Ratios

Meshing of areas with quadrilateral elements and volumes with brick elements is available in certain cases. The most limiting restriction is that only the standard region shapes (four keypoints on areas, eight keypoints on volumes) are allowed. Further, the number of element divisions requested must match on opposing sides of areas (figure 18). The area corner angles must be reasonable for quadrilateral or brick elements. The mesh is mapped onto the natural coordinates of the areas and volumes (figures 19 and 20).

Matching Divisions

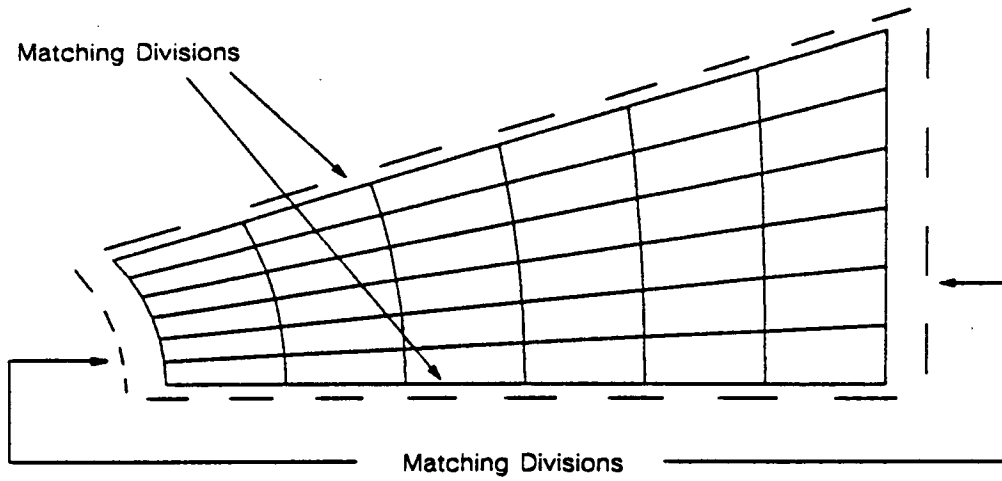Matching Divisions

Figure 18   Matching Divisions on Areas Required for Quadrilateral or Brick Meshing
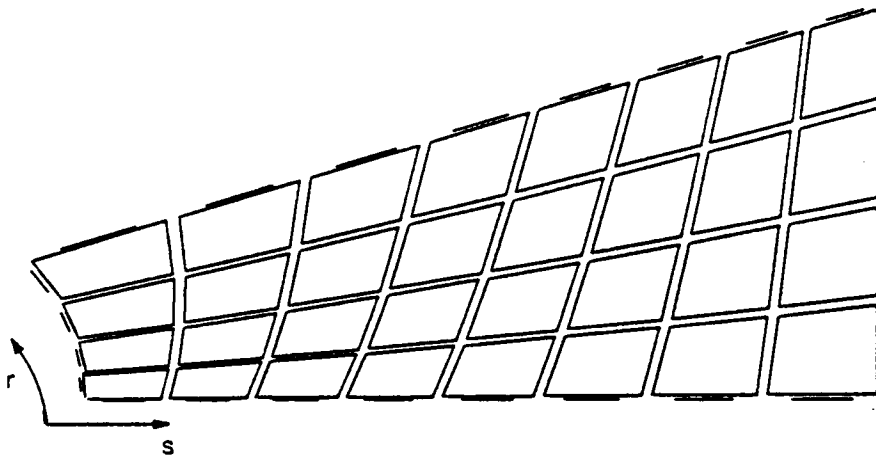
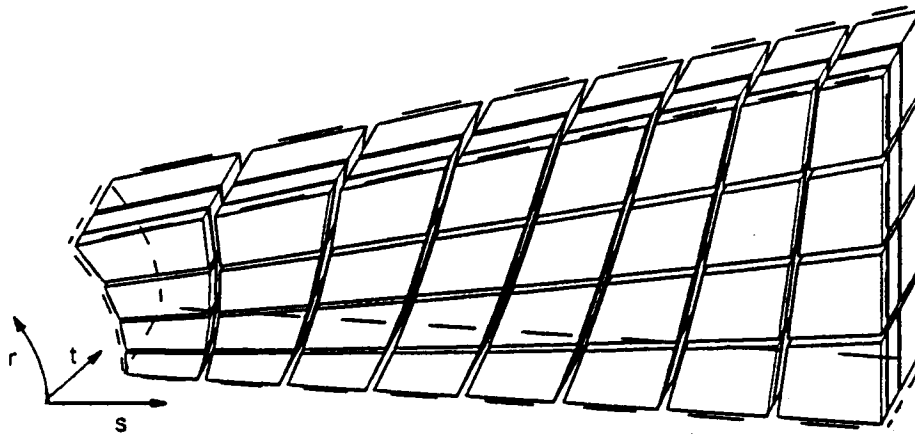Figure 19   Quadrilateral Element Meshing of an Area

Figure 20   Brick Element Meshing of a Volume

Meshing with triangles is available for all areas, regular or not. Meshing with tetrahedra is available for all volumes, regular or not. The elements of choice are the 6-noded triangular solid or shell (figure 21) and the 10-noded tetrahedral solid (figure 22). ANSYS has these elements available for stress, thermal, electro-magnetic, or multi-field analysis. For planar, axisymmetric, or shell applications, 6-noded triangles are good performers, giving results of equal or superior quality for equal edge divisions when compared to 4-noded or 8-noded quadrilaterals. For 3-dimensional solid applications, 10-noded tetrahedra perform well. (This element is a theoretically consistent, completely conforming element which passes the patch test. Because tetrahedral meshes are rarely symmetric, however, this element can develop localized spurious deformation modes. For this reason, some theoreticians have refused to bless this element for general use. This is really bad news if their fear is justified, since tetrahedral meshing is the only reasonable approach to automated meshing of arbitrary 3-dimensional shapes. No conclusions can be reached, however, until the stress analysis community has had ample opportunity to gain experience with tetrahedra.) ANSYS uses the same algorithm for triangular meshing of areas and tetrahedral meshing of volumes: an initial mesh is formed without regard to region shape and is then repeatedly improved by operations which divide or combine elements, until all elements are nicely shaped or until no operations available will improve the situation. This iterative scheme is computationally intensive, but is highly reliable and produces well distributed meshes of well shaped elements.
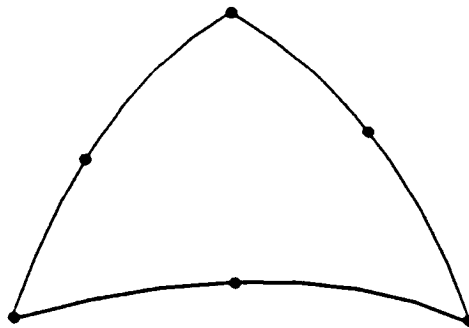

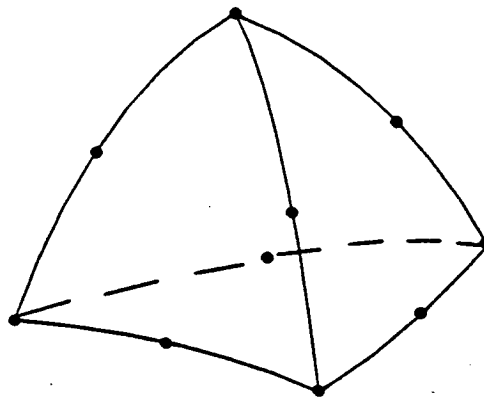Figure 21   6-Noded Triangle Element

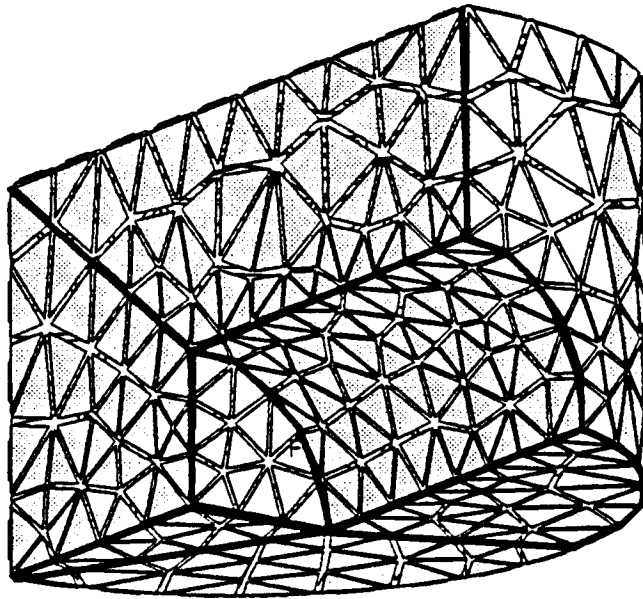Figure 22   10-Noded Tetrahedron Element

Figure 23   Triangular Mesh on the Exterior of a Volume
Prior to Tetrahedron Meshing

We at SASI have been asked on several occasions why we do not mesh areas with mixtures of quadrilaterals and triangles, or mesh volumes with mixtures of bricks, wedges, and tetrahedra. First, there is little evidence to suggest that such mixed meshes are likely to perform any better than meshes consisting entirely of triangles or tetrahedra. Second, the algorithms to produce such meshes appear to be at least as complex and compute intensive as the triangle and tetrahedron algorithms, if they are to check element shape as thoroughly as they should. Finally, connecting brick elements to wedges and tetrahedra is not a straightforward process if one wishes to avoid displacement incompatibilities.

Meshing of adjacent areas or adjacent volumes in ANSYS will always produce compatible and properly interconnected finite element meshes. This is possible because the triangle and tetrahedron meshing algorithms used do not have the "authority" to alter the exterior of the mesh of a region. As shown in figure 23, the tetrahedral meshing of a volume starts with a fixed exterior triangular mesh, which cannot change.

6. BOUNDARY CONDITIONS

The following boundary conditions may be defined directly on an ANSYS solid model.

imposed displacements at keypoints   (stress analysis)
imposed temperatures at keypoints   (thermal analysis)
imposed voltage at keypoints   (electrical analysis)
imposed magnetic potential at keypoints   (magnetic analysis)

(constraints can be interpolated over attached line segments, areas, and volumes)

temperatures at keypoints   (stress analysis)
heat generation rates at keypoints   (thermal analysis)

(can be interpolated over attached line segments, areas and volumes)

13

applied forces at keypoints	(stress analysis)
applied heat input at keypoints	(thermal analysis)
applied current flow at keypoints	(electrical analysis)
applied magnetic flux at keypoints	(magnetic analysis)
pressures on line segments	(2-D stress analysis)
convections on line segments	(2-D thermal analysis)
symmetry / antisymmetry on line segments	(2-D solid or 3-D shell stress analysis)
pressures on areas	(3-D stress analysis)
convections on areas	(3-D thermal analysis)
symmetry / antisymmetry on areas	(3-D stress analysis)

Boundary conditions may be defined before or after finite element meshing, and can be displayed on the solid model. They will be transferred to the finite element model automatically when needed. (The transfer can be forced earlier if the user wishes to display them on the finite element model.)

Even if boundary conditions are not applied directly to an ANSYS solid model, they can be conveniently applied to a finite element model created by the solid modeler. Nodes and elements associated with various mesh entities can be activated or deactivated as desired, making it easy to specify where constraints or loadings belong.

7. ANSYS INTERFACE WITH CAD SYSTEMS

ANSYS accepts keypoint and line segment information from a number of other solid modeling systems (see Table 1). A user can use this data to create areas and/or volumes and a finite element model.

Table 1

Translations within ANSYS	Other Systems Having Some Interface with ANSYS		
IGES	ADAMS-DRAM	CIS-MEDUSA	PATRAN
MEDUSA	ADVANTAGE	DIAD Solid Modeler	PDGS
FEMVIEW	ANVIL	ENGINEER WORKS	PROF. CADAM
FEMGEN	APPLICON	EUCLID	ROMULUS-D
NASTRAN	AUTOCAD	FEMAS	SOLUTION 3000
SUPERTAB	AXXYZ	GRAFTEK	UNIGRAPHICS
	CADAM	ICAD	VERSACAD
	CADKEY	INTERGRAPH	
	CATIA	ME30	
	CDS-4000	M.E. Workbench	
	CIMLINK	MOLDFLOW	

Even though ANSYS can accept nodes and elements created by other systems, we believe that in most cases the user will be better off doing the finite element meshing step within ANSYS. First, we have seen evidence (finite element models from other systems) that not all developers of meshing software know what constitutes a good analysis model. Badly shaped elements may give poor quality analysis results. (It is far better to inform the user that meshing is not possible with

the data supplied than to produce an unacceptable mesh.) Second, if the user has meshed in another system, he or she may be reluctant to make any alterations to the model which may be indicated by initial analysis results. Third, the required mesh may by load dependent. Finally, nodes and elements brought into ANSYS from another system will not be associated with the solid model, and boundary condition manipulation will be difficult.

8. FUTURE DEVELOPMENT PLANS

In the short term (Revision 4.4, 1988), we plan improvements in the command structure for defining ANSYS solid models. We hope to improve the speed and reliability of our meshing algorithms. We plan to allow definition of contact surfaces. We plan to improve our interfaces with other software packages.

In the long term (Revision 5, 1990), we want to address some or all of the following.

- mapping analysis results back onto the solid model
- adaptive mesh refinement
- improved curved surface accuracy
- improved user interface

9. EXAMPLES

Figures 24 through 31 show several examples of ANSYS solid models and resulting finite element meshes. Table 2 shows the various statistics for these models.

Table 2
Example Statistics

| Model	Number of Commands	Number of Elements		Elapsed time* for creation of solid model & finite element model	Computer
Block with two holes	159	3047	tetrahedra	183 minutes	VAX 11/780
Helix	62	2856	tetrahedra	160 minutes	Prime 9950
Pawn	60	819	triangles	19 minutes	MicroVax
Knight	351	2427	tetrahedra	318 minutes	MicroVax
Gear	279	1136	tetrahedra	31 minutes	Prime 9950
Gear Submodel	186	2684	tetrahedra	138 minutes	Prime 9950
Turbine Spacer	403	1224	tetrahedra	75 minutes	Prime 9950

*CP times are nearly identical
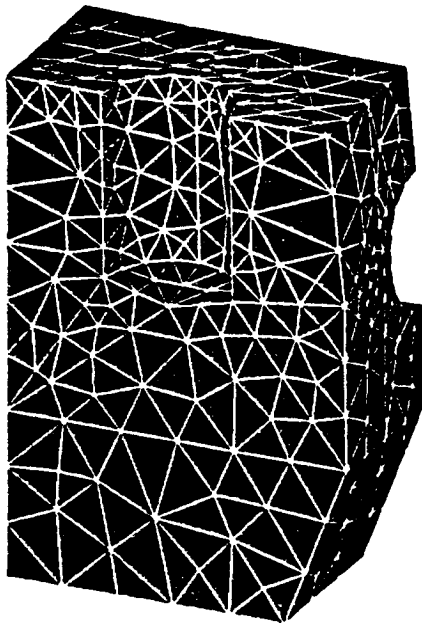
Figure 24   Tetrahedron Model of Block with Two Holes

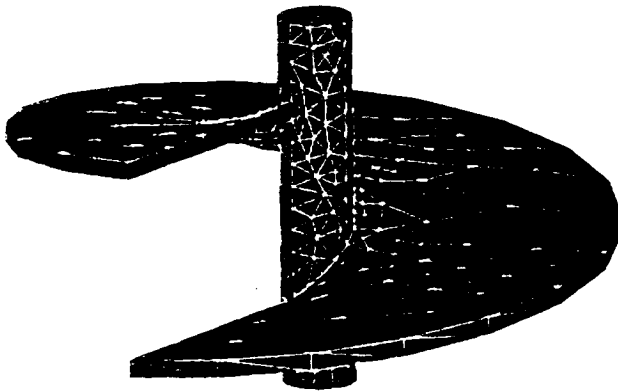Figure 25   Detail of Tetrahedron Model – Block with Two Holes

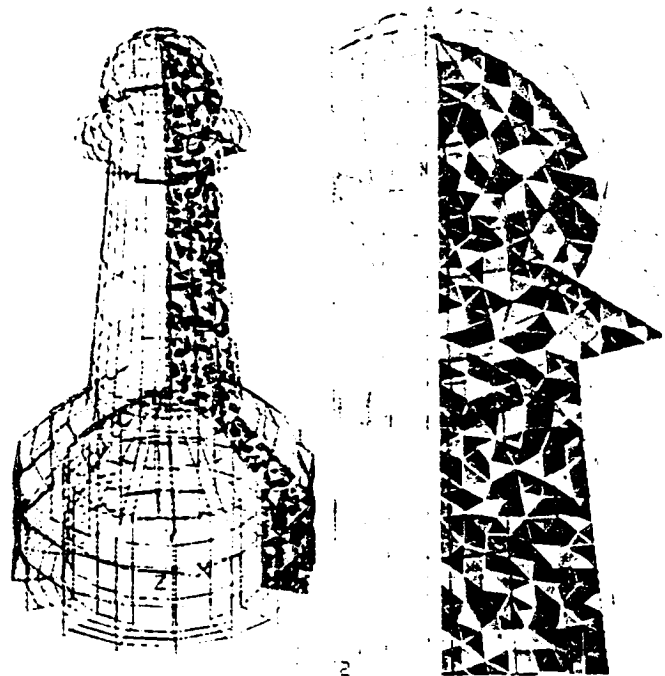Figure 26   Tetrahedron Model of Helix

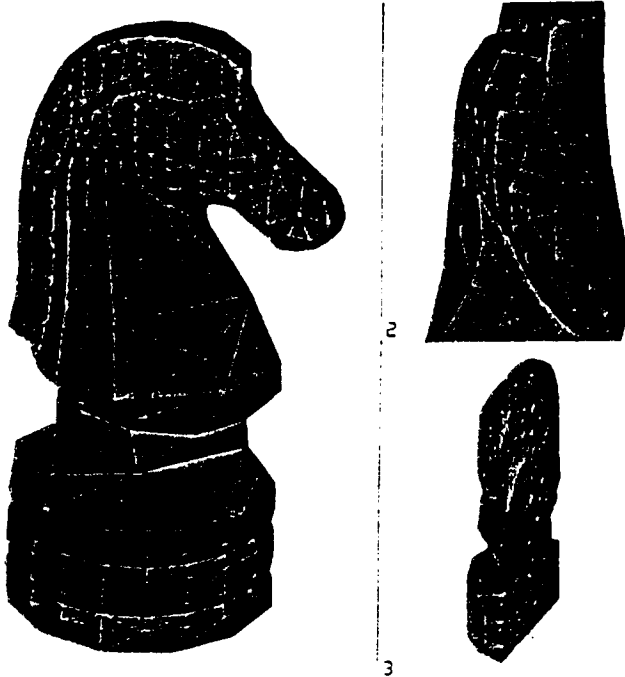Figure 27   Triangle Model of Pawn
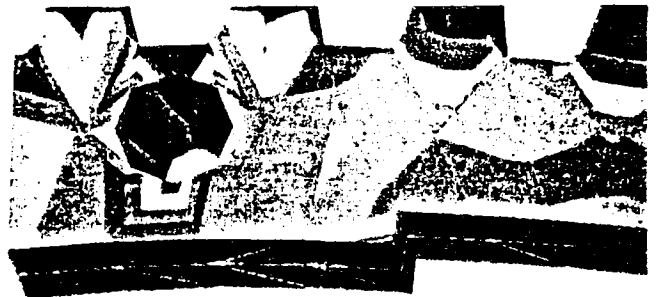
Figure 28   Tetrahedron Model of Knight

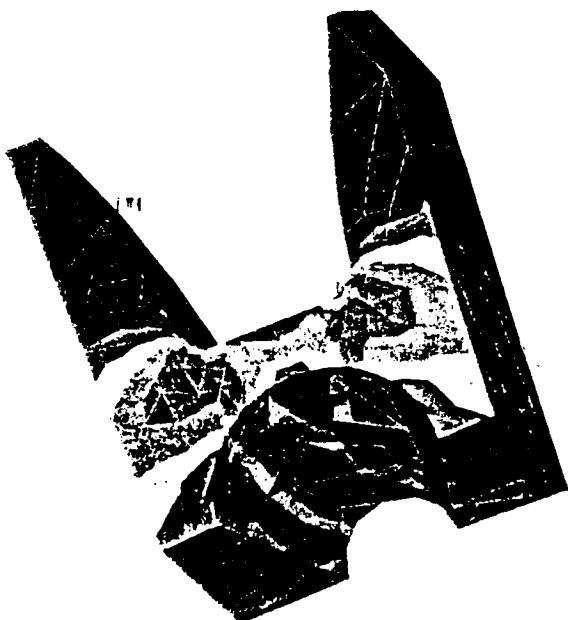Figure 29   Stress Contours on Tetrahedron
Model of Gear

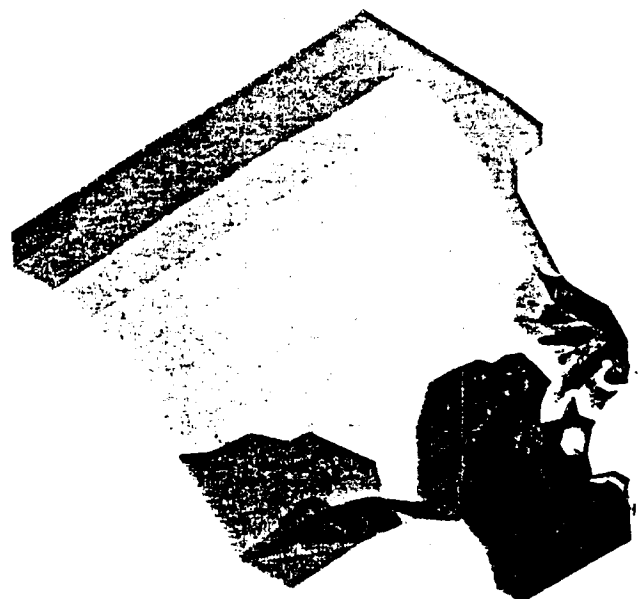Figure 30   Stress Contours on Tetrahedron
Submodel of Gear

Figure 31   Stress Contours on Tetrahedron
Model of Turbine Spacer